

Stress Analysis of Fuel Access Cut out of the bottom skin of a transport aircraft

Sartaj Patel¹, Mahesha.K², Harish E.R.M³

P G student, Department of Mechanical Engineering, Acharya Institute of Technology, Bangalore, India¹

Professor and Head, Department of Mechanical Engineering, Acharya Institute of Technology, Bangalore, India²

P G student, Department of Mechanical Engineering, Acharya Institute of Technology, Bangalore⁴, Karnataka, India³

Abstract: Aircraft is symbol of a high performance mechanical structure with a very high structural safety record. In this an attempt had made to find the maximum tensile stress location in the bottom skin of transport aircraft by considering the behaviour of side spars for bending and ribs for transverse load. The component wing box with the access cut out in the bottom skin is considered for the analysis in this the behaviour of each rib and side spars is calculated or known. The maximum tensile stress location is identified. This location is the stress concentrated areas. In this location should be identified and the crack can be avoided.

1.INTRODUCTION

Airframe engineers view any cut outs in airframe structures with disfavour because the necessary reinforcement of the cut out increase costs and adds weight to the overall design. In addition, the design and sizing of cut outs is a difficult process since it is an area of stress concentration, a problem area for both static and fatigue strength and there is insufficient design data.

Cut outs are essential in airframe structures to provide the following;

- Fuel access cutout at the bottom skin of wing and fuselage.
- Landing gear opening and retracting at the bottom skin of the wing or fuselage.
- Lightening holes in webs.
- Accessibility for final assembly and maintenance (e.g., manholes in wing lower surfaces, crawl holes in wing ribs, etc.).
- Inspection for maintenance (medium sized cutouts called hand holes).
- Window cutout in fuselage.

In the current work, the component considered is the Fuel access cut out of wing bottom skin which provides the passage for fuel access in wing structure of the light weighted transport aircraft.

During flight, the lower part of the wing experiences a tensile stress and the upper part experiences a compressive stress. The current case considers the bottom or lower part of the wing, which experiences tensile stresses. In order to withstand the bending of the wing section due to transverse loads acting on the wing, the wing box is provided with integrated stiffeners. Cut out which is intended to provide passage for fuel access comprises of auxiliary holes around the small cut out. Discontinuities or flaws in any structure leads to high stress concentration at that region. Here, cut-out with auxiliary holes will be the critical region. These are probable locations for fatigue crack initiation. So stress analysis of that wing box is necessary.

II. GEOMETRICAL CONFIGURATION

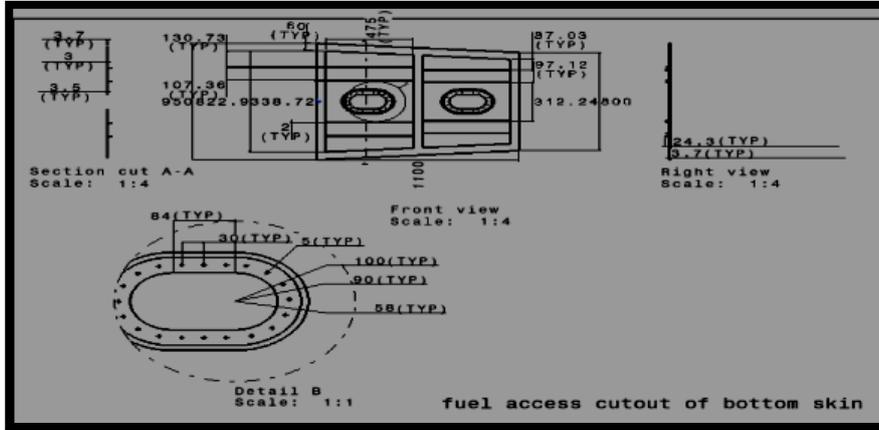


Fig.2.1 Bottom skin of the fuel access cutout



Fig 2.2: Top Skin of Fuel Access cut out

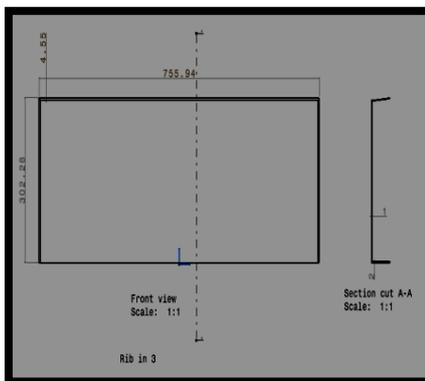


Fig 2.3: Rib of Fuel Access cut out

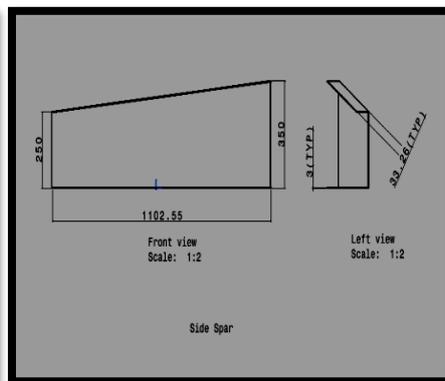


Fig 2.4: Spar of Fuel Access cut

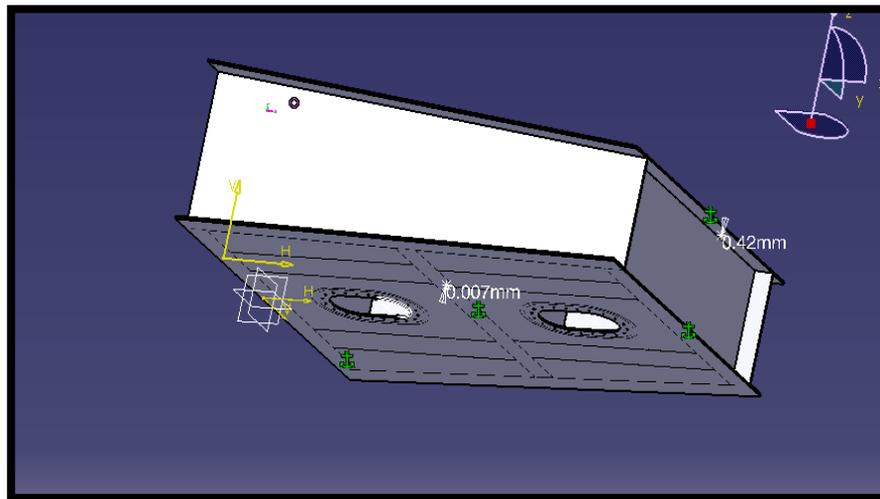


Fig 2.5 CAD part of Fuel Access cut out of Wing Bottom skin

Fuel access cut out of wing bottom skin consist in the bottom skin^{2.1} two capsule holes of large diameter for the access of the fuel from the tank. Small (auxiliary) holes around the cut outs of the tank are made for the revitation, revetes for the fixing of required accessories and also consists of the two sides spars for the bending load carrying as shown in the figure 2.4 and one top side plate^{2.2} for the bending, it has three ribs across the position in the fuel tank for the transverse load carry as show in figure 2.3. A CAD part of assembled section is shown below figure3.5. Bottom skin and top plates consists of stringers in longitudinal direction to increase the strength of plates, there are shown on the plates as above.

III.MATERIAL SPECIFICATION

Selection of aircraft materials depends on any considerations, which can in general be categorized as cost and structural performance. Cost includes initial material cost, manufacturing cost and maintenance cost. The key material properties that are pertinent to maintenance cost and structural performance are: density, stiffness, strength, durability, damage tolerance, corrosion.

The material considered for the lug part of the structure is Aluminium Alloy – 2024-T351, with the following properties.

1. Young's Modulus, $E = 72400\text{N/mm}^2$
2. Poison's Ratio, $\mu = 0.3$
3. Ultimate Strength, $\bar{\sigma}_u = 503.7\text{N/mm}^2$

IV.LOADS ON THE FUEL ACCESS CUT OUT OF WING BOTTOM SKIN

Most of the wing bending is carried by the spars in the wing structure. The maximum bending moment occurs at the root of the spar where wing and fuselage components will be attached to each other. The load calculation for the Fuel access cutout of wing bottom skin is described in the next section.

A: Load calculation for the fuel access cutout of wing bottom skin

The load experienced by the wing box is obtained as follows:

Weight of the Aircraft structure =15696N

Load factor =3.0 g

Factor of safety = 1.5

Total load = $15696 \times 4.5 = 70632\text{N}$

Lift load acting on the wing = 80% of total load = $70632 \times 0.8 = 56505.6\text{N}$

Load acting on each wing = $56505.6 / 2 = 28252.8\text{N}$

Span of the wing = 3000 mm

Location of resultant load from the root of the wing = 1250 mm

Bending moment at the root = $2880 \times 1250 = 3.53 \times 10^7 \text{N-mm}$

Load to be applied at the end of the wing box considered for the analysis = $3.53 \times 10^7 / 168$

= 20959.05N

Total Edge length of the component where load is applied = 1851.4 mm

Total UDL load applied for the component = $20959.05 / 1851.4$

= 11.31 N/mm

V.FINITE ELEMENT ANALYSIS

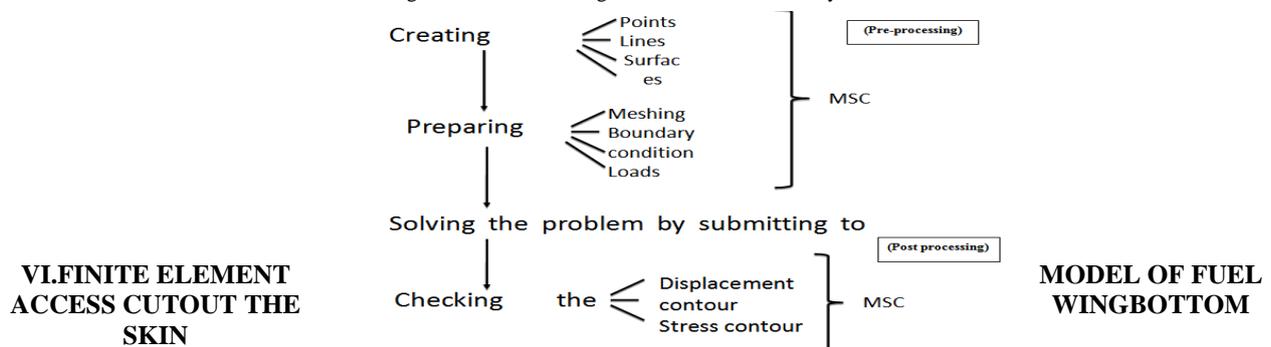
A: Introduction to FEA approach

The finite element method (FEM) is a numerical technique for solving problems which are described by partial differential equations or can be formulated as functional minimization. A domain of interest is represented as an assembly of finite elements. Approximating functions in finite elements are determined in terms of nodal values of a physical field which is sought. A continuous physical problem is transformed into a discretized finite element problem with unknown nodal values. For a linear problem, a system of linear algebraic equations should be solved. Values inside finite elements can be recovered using nodal values.

B: The different stages of finite element analysis

The software used for the analysis of the Landing gear lug attachment joint in an airframe is MSC.Patran & MSC.Nastran. The stages involved in FEM are shown in the figure below

Fig.5.1 The different stages of Finite Element Analysis.



As per the design calculations from the previous section the dimensions of the fuel access cutout of the wing bottom skin. All other dimensions of the complete assembly of the structure are as per the description provided in the previous section in the problem definition chapter. A finite element model is the complete idealization of the entire structural problem including the node location, the element, physical and material properties, loads and boundary conditions. The purpose the finite element model is to make a model that behaves mathematically as being modelled and creates appropriate input files for the different finite element solvers. In Finite Elements libraries, selected 4 Nodded

QUADRILATERAL Shell Element (QUAD4). In this Geometrical model for available surface area, chosen for formulation of FE Model, reason was flow of displacement and stiffness.

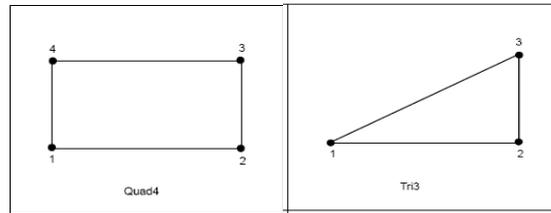


Fig.6.1 3Noded TRIA Shell Element And 4Noded Quadrilateral Shell element.

FE model of the wing fuselage Lug attachment bracket is as shown in Figure 5.4 Meshing is carried out by using CQUAD4 and CTRIA3 shell elements. Triangular elements are used for the transition between the coarser mesh to finer mesh.

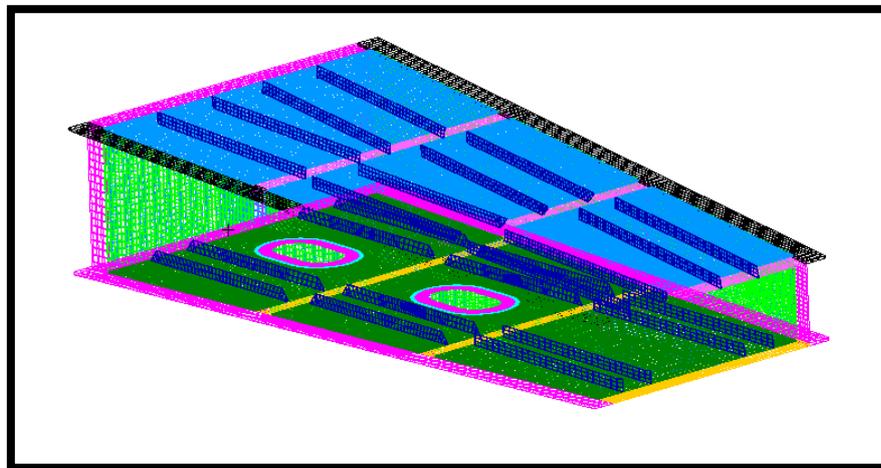


Fig.6.2 Finite element (FE) model of the fuel access cutout of the wing bottom skin.

VII.LOADS AND BOUNDARY CONDITIONS

The loads and boundary conditions along with the finite element model are shown in the figure 7.1. A load 11.31 N/mm of is introduced at one end of the spar beam. This load will essentially create the required bending moment at the root.

The top and bottom auxiliary holes of the Fuel access cutout of wing bottom skin are constrained with all six degrees of freedom at the semicircular circumferential region.

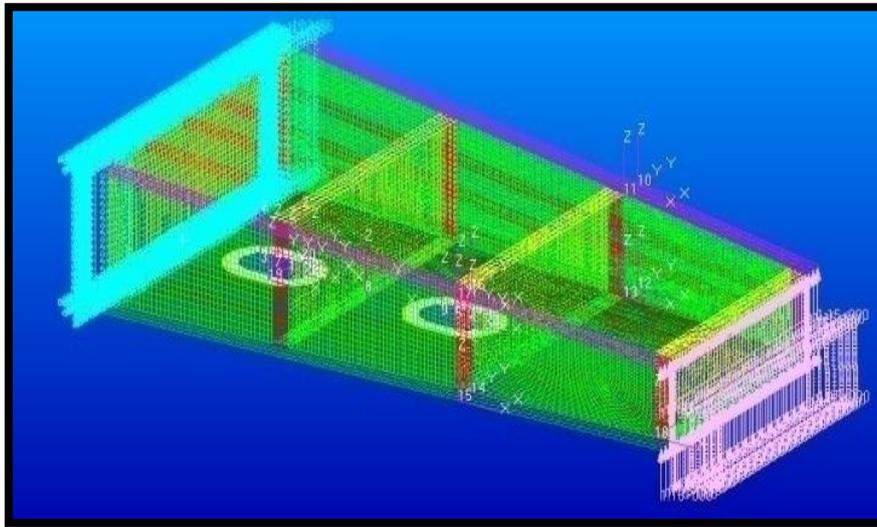


Fig.7.1 Load and Boundary conditions applied to the fuel access cutout of the wing bottom skin.

VIII.FINITE ELEMENT MODELLING AND STRESS ANALYSIS OF THE FUEL ACCESS CUTOUT OF WING BOTTOM SKIN

The stress values at the Fuel access Cutout of wing bottom skin and the displacement contours are shown in the figures 8.1 (figure 8.5 shows the close up view of maximum stress at hole) and figures 8.6 respectively. A maximum stress of 108.891N/mm^2 is observed at the small hole (auxiliary) section. Ultimate strength of Aluminium Alloy – 2024-T351 is 503.7N/mm^2 . Here max. Stress is less than the ultimate strength of structure; therefore we can say structure is safe for applied load. (I.e. Fuel access Cutout of wing bottom skin edge load is $20.954 \times 10^3\text{N}$).

When the Stress is greater than or equal to the ultimate strength of structure than material is going to fail. A maximum displacement of 9.55mm at the free end of the structure can be observed from the displacement contour in the figure 8.6.

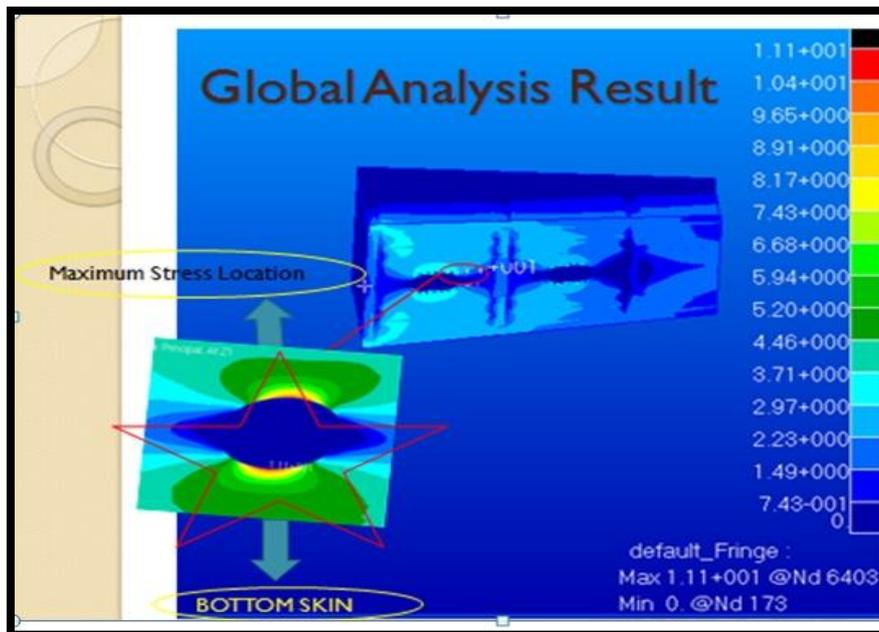


Fig 8.1 Figure Showing Maximum Stresses of the fuel access cutout of the wing bottom skin.

Table 8.1 convergence requirements

No. of iterations	$\sigma_{\max}(\text{N/mm}^2)$
0	0
1	103.986
2	106.929
3	108.899
4	108.899
5	108.899

Fig 8.2 Graph of the Final Convergence of the Stress Values

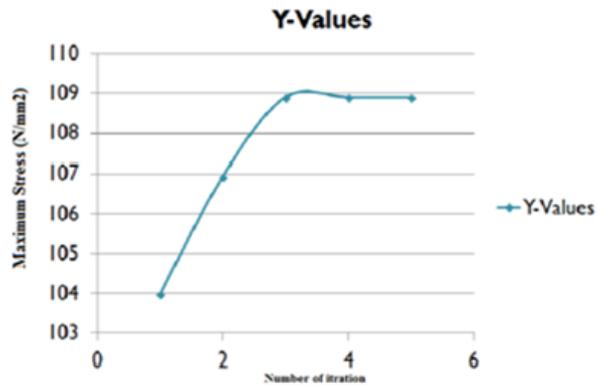


Table 8.1 shows the convergence criteria of Fuel access cutout of a wing Bottom skin. It useful to convert from coarser mesh to fine mesh. In each and every iteration number of element will increases gradually, due to increasing in the number of elements, size of the element will decreases and stress concentration will also reach the exact value of the structure. At certain level it will be constant as shown in the graph 8.2. In this process for iterations have got same answer as shown in table 8.1. By using these iterations we can select the accurate maximum stress in Fuel access cutout of a wing Bottom skin.

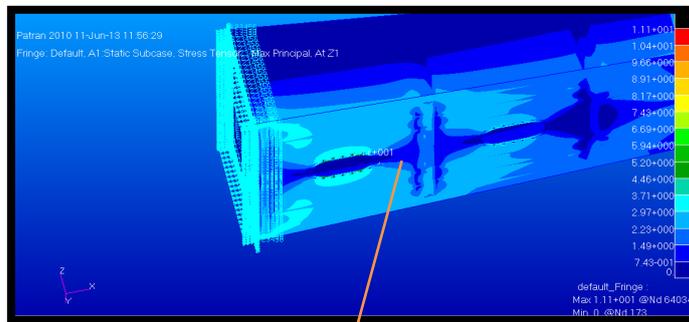


Fig.8.4 Stress contour at the the fuel access cutout of the wing bottom skin.

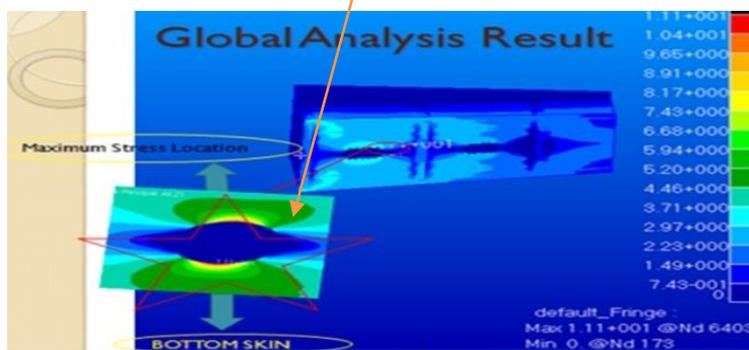


Fig 8.5: close up view of the stress contour the fuel access cutout of the wing bottom skin.

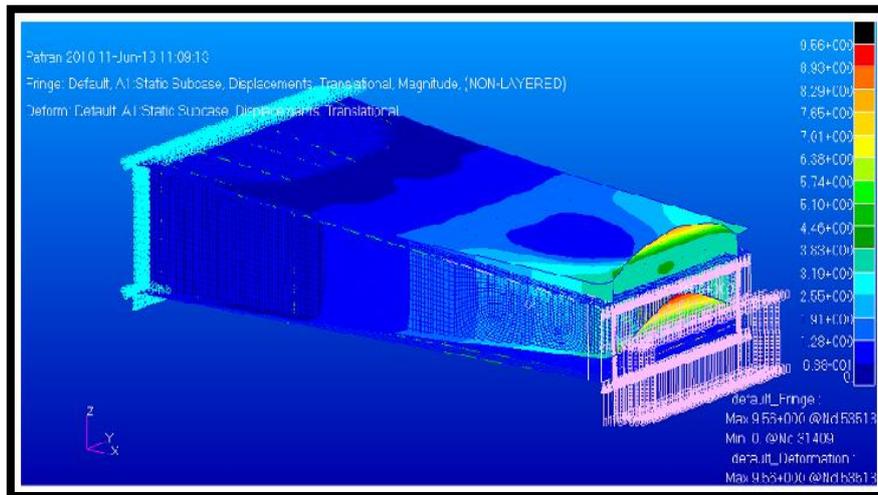


Fig.8.6 Max. Displacement in Fuel access cutout of wing bottom skin

9. RESULTS AND DISCUSSIONS

The stress contour indicates a maximum stress of 108.891N/mm^2 at fuel access cutout of wing bottom skin as shown in the figure 8.1. The maximum stress value obtained is within the yield strength of the material. The point of maximum stress is the possible location of crack initiation in the structure due to fatigue loading.

10. CONCLUSIONS

- Stress analysis of the fuel access cutout of wing bottom skin is carried out and maximum tensile stress.
- FEM approach is followed for the stress analysis of the fuel access cutout of wing bottom skin .A validation for FEM approach is carried out by considering a plate with a circular hole.
- Maximum tensile stress of 108.891N/mm^2 is observed in the fuel access cutout of wing bottom skin.
- Several iterations are carried out to obtain a mesh independent value for the maximum stress.
- A fatigue crack normally initiates from the location maximum tensile stress in the structure.
-

ACKNOWLEDGEMENT

The analytical work was conducted Bangalore Aircraft Industries Private Limited, the author gratefully acknowledge their support The author also would like to thank K E Girlish(DIRECTOR, BAIL) for providing expert guidance and advice for the analytical work. Addition the author would like to thank theto Dr. Mahesha.K. Professor and Head of the Department, Mechanical Engineering, Acharya Institute of Technology, Bangalore.

REFERENCES

- [1] National transport safety board, Washington DC, "Aircraft accident report-Fairchild F-27B". N4905, Pedro Bay, Alaska; December 2, 1968.
- [2] Richard A Everett Jr., "The effects of load sequencing on the fatigue life of 2024-T3 Aluminium alloy". International Journal of Fatigue, Volume 19, Issue 93, June 1997, pages 289-293.
- [3] Dr.M.Neubauer, G.Gunther, "Aircraft loads", *Aging Aircraft Fleets: Structural and Other Subsystem Aspects*, Bulgaria. 13-16 November 2000, pages 9:1-19.
- [4] A.K. Vasudevan, K. Sadananda and G. Glinka, *Critical parameters for fatigue damage*. V International Journal of Fatigue, Volume 23, 2001, pages 539-55
- [5] Weicheng Cui, "A state-of-the-art review on fatigue life prediction methods for metal structures". Journal of Marine Science and Technology, January 7, 2002, pages 7:43-56.

BIOGRAPHY

Mr. Sartaj Patel obtained his B.E Mech from KCT Engineering College, Gulbarga in 2010 and is P G student (M Tech Machine Design) at Acharya institute of Technology, Bangalore. He is a Member of American Society of Mechanical Engineers (ASME).



Dr. Mahesha K, ME., Ph.D, presently working as a Professor and Head of the Department of Mechanical Engineering, Acharya Institute of Technology, Bangalore. He holds Life Time Membership of Indian Society for Technical Education (LMISTE) and Material Research Society of India (MRSI). He has published several papers in International Journals. His research area is in advanced materials and Damping characteristics of materials.



Mr. Harish E.R.M obtained his B.E Mechanical from BIET, Davangere, in 2011 and is a P G student (M Tech Machine Design) at Acharya institute of Technology, Bangalore. He is a Member of American Society of Mechanical Engineers (ASME).